COMPUTATIONAL ANALYSIS OF REVERSE FLOW COMBUSTOR FOR SMALL GAS TURBINE APPLICATION

BY

Muthuselvan G, Muralidhara H S, DineshKanth T P, LakshmiNarayana Y

Combustion Lab, Propulsion Division,
CSIR - NAL, Bangalore -560037.

Abstract: The present work reports a computational study of flow through reverse flow combustor for various values of air to fuel ratios (75,100,125,150 and 175). The reverse flow combustor with eight fuel injectors is modeled using solid works software. A 45° section of reverse flow combustor is considered for present study. Three dimensional flow through the reverse flow combustor have been simulated by means of computational fluid dynamics (CFD) using the Ansys Fluent code. The objective of this work is to understand the flow field characteristics of reverse flow combustor and to study the effect of increased values of air to fuel ratios on primary zone temperature distribution and performance of reverse flow combustor. The flow field of reverse flow combustor is well captured by solving appropriate governing equations using the SIMPLE algorithm. Turbulence has been modeled using K-e turbulence model. Hot flow simulations of different fuel flow rate cases (Air and Jet-A mixture) are simulated by non-premixed combustion model. It has been observed that increase in fuel flow rate shifts hot flow region away from swirler exit.

Introduction

There are many types of combustor configuration available for the use in small gas turbine turbo machinery. But the reverse flow combustor offers many advantages when compared to other types. Reverse-flow combustor can reduce the length of the gas turbine and make transportation much easier. Reduced length will allow single shaft to sit on two bearings instead of three. This will reduce the vibration and maintenance problems. The reverse-flow layout effectively uses the air flow to cool down the combustor liner. The absorbed heat by the air is returned back to the system. In other words, the reverse-flow combustor cooling process is actually a preheating process of the air. The reverse-flow process also allow warmer air to serve as the dilution air to control the NOx formation instead of using other energy to preheat the dilution air or use cold air which could quench the flame and produce CO.

Considerable amount of numerical work has been done to understand the flow behaviour of reverse flow combustor. D.S. Crocker and C.E.Smith (1995) performed numerical study to understand the effect of advanced dilution hole concept (injecting dilution air jets with high circumferential component) on pattern factor. From the numerical work, the authors found optimum dilution hole spacing, jet angles and mass flow split to improve the quality of temperature distribution at exit. G. Boudier, L.Y.M Gicquel and T.J.Poinsot (2008) performed numerical work using unstructured compressible LES solver. The authors studied the effect of mesh resolution on reacting flows in complex geometry combustors. Results show that the mean temperature, reaction rate, and velocity fields are almost insensitive to the grid size. The RMS field of the resolved velocity is also reasonably independent of the mesh, while the RMS fields of temperature exhibit more sensitivity to the grid. Jianli Chen, Chaoqunie and Weiguang Huang (2008) performed numerical investigation on stagnation point reverse flow combustor. The numerical results indicated that increase in fuel and air mixture injection velocities increases the CO emissions, decreases flame temperatures and NO_x emissions. Also the authors observed that as the equivalence ratios increase from 0.5 to 1, the NO_x emissions always increase. However, the CO emission decreases first, reaching the minimum value at equivalence ratio of 0.58, and then increases. Satish Undapalli, Srikant Srinivasan, Suresh Menon (2009) performed numerical analysis on stagnation point reverse flow (SPRF) combustor. The flow features and the combustion characteristics of both premixed and non-premixed modes are studied using LES. It is shown that premixing with hot products is the process that enables stable operation at a very lean equivalence ratio.

Objectives

1. To predict the flow through reverse flow combustor.

2. To study the effect of overall air to fuel ratio (75,100,125,150,175) on primary zone temperature distribution, total pressure loss and circumferential pattern factor of reverse flow combustor.

Assumption and boundary conditions

1.The flow is assumed to be steady and turbulent.

2.The geometry is assumed to be axisymmetric.

3. Mass flow inlet boundary condition is used at air and fuel inlet with turbulent intensity of 1%.

4. Pressure outlet condition is used at exit with turbulent intensity of 7%.

Geometry and grid generation

The 45 degree section of reverse flow combustor model used in the present study is shown in figure – 1. It consists of several features like Swirler, Dome with four rows of cooling holes with T strip, Atomizer with fuel injection pipe, Primary and dilution holes, Outer liner cooling holes, and inner liner with cooling holes. The liner assembly with fuel injection pipe is shown in figure – 2. Combustor model is generated in solidworks software. The combustor has 8 fuel injectors. So the simulative domain adapted is 45 degree of the whole combustor. The three dimensional grid used in the present analysis is shown in figure – 3. The grid was generated using Ansys-meshing software. The total number of nodes is 36,45,362. To effectively capture temperature contour downstream of swirler fine mesh is generated in the primary and dilution zone of combustor. Coarse mesh is generated in the stagnation zone of the combustor. Very fine mesh is generated near cooling holes.

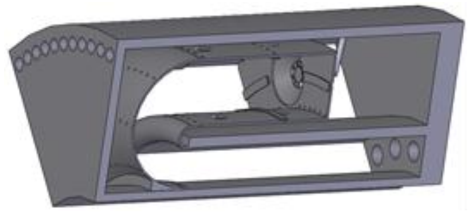

Figure – 1 Computational domain

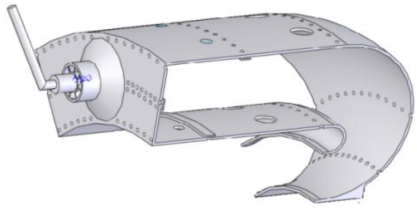

Figure – 2 Liner with atomizer and swirler assembly

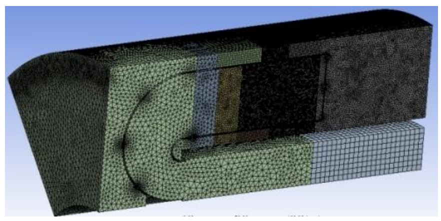

Figure – 3 Meshed reverse flow combustor

Results and discussion

The present analysis started with cold flow investigation. Next, gaseous Jet-A fuel is introduced from fuel inlet. Finally fuel flow rate is decreased to simulate different air to fuel ratio from 75 to 175 insteps of 25 with constant air flow rate.

Cold flow

The figure – 4 shows mid plane (in-line with atomizer) velocity contour of cold flow case. It shows penetration of air through various holes in liner (Primary, dilution and cooling holes). Since air is admitted in to the combustor through holes; velocity of air at inlet is higher as shown in figure – 4. The figure – 5 shows velocity vector plot of transverse plane, which is very close to swirler exit. It shows entry of air into the core through swirler and four rows of dome cooling holes. Figure – 6 and 7 shows entry of air through primary holes into the liner from outer and inner annulus respectively. The injection of air through primary holes (both inner and outer) forms small vortices around each jet. These vortices were well captures by fine mesh present near the holes. The predicted percentage of mass flow distribution through each zone of combustor for hot and cold flow is shown in table – 1. There is not much change in flow distribution between hot and cold flows.

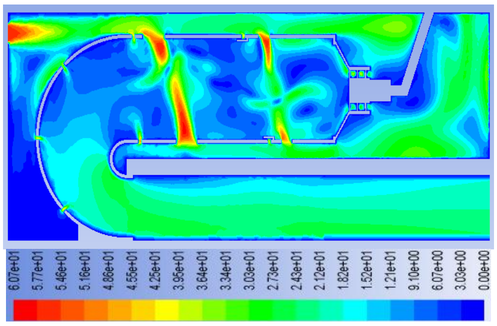

Figure – 4 Mid plane contour of axial velocity of cold flow case

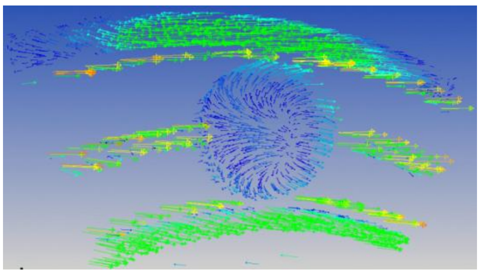

Figure – 5 Velocity vector plot at swirler exit

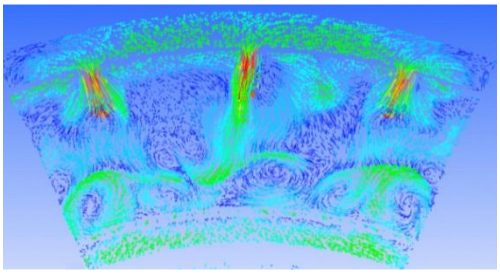

Figure – 6 Velocity vector plot at primary zone (outer annulus

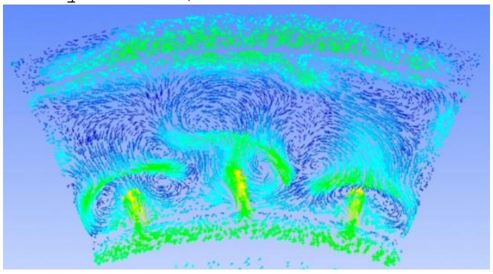

Figure – 7 Velocity vector plot at primary zone (Inner annulus)

Name of zone	Percentage of mass flow	
	Cold flow	Hot flow
Core flow	20.5	20.75
Inner Primary	7	7.2
Outer Primary	11	11.1
Inner Dilution	21	20.63
Outer Dilution	25.5	25.65
Liner Cooling	15	14.67

Table – 1 mass flow distribution of both cold and hot flow cases.

Hot flow

Figure-8 shows the mid plane temperature contour of air to fuel ratio = 125 case. It depicts hot flow region located in the primary zone of combustor. Cold air jet entering through primary holes perfectly mixing with hot gas. Some amount of unburned fuel coming from primary zone burns with primary air jet and forms small region of high temperature zone which is located downstream of primary holes. Figure – 9 shows side plane temperature contour. This side plane is located in between two injectors. By comparing figure – 8 and 9, it can be observed that high temperature zone is slightly shifted towards swirler exit in side plane contour. The reason for this shifting is because of more fuel availability in the center region than side plane region. Presence of higher fuel near swirler exit forms locally rich mixture, which forms low temperature region. Figure – 10 shows temperature contour in the transverse plane, which is located between of swirler exit and primary hole. Relatively low temperature zone is observed in the center of transverse plane. The corrugated region of high temperature zone is observed near the edges of upper and lower liner. This corrugation is mainly because of presence of dome cooling holes. Figure – 11 shows temperature contour in the transverse plane, which is located at primary hole entry. Unburned fuel and partially burned fuel coming from core region mixes with primary air and forms high temperature region in the center, so relatively high temperature zone is observed in the center.

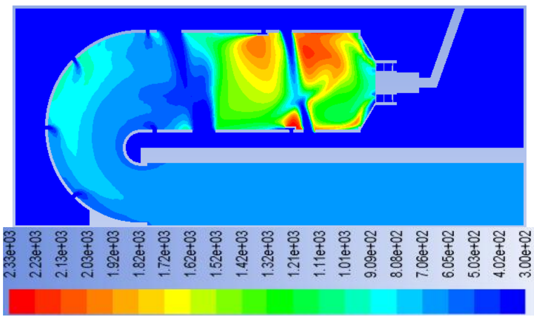

Figure – 8 Mid plane temperature contour of hot flow

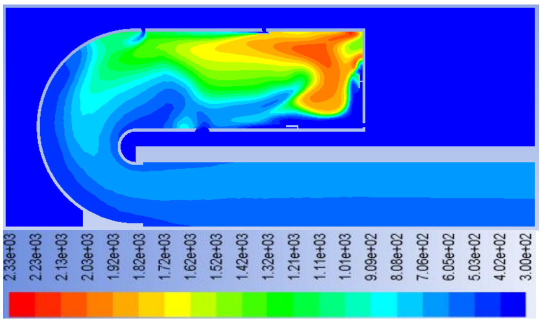

Figure – 9 Side plane temperature contour of hot flow

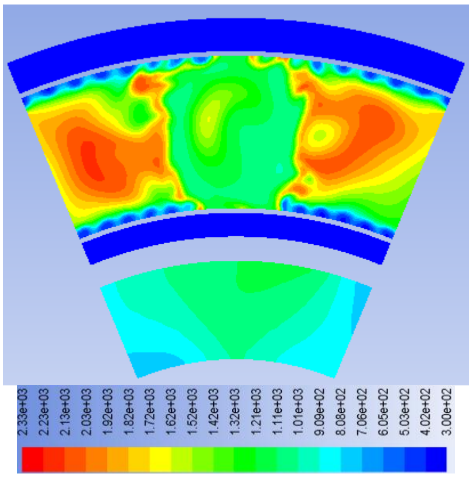

Figure – 10 Transverse plane temperature contour at swirler exit

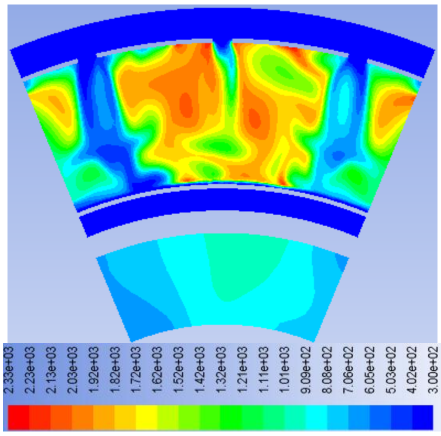

Figure – 11 Transverse plane temperature contour at primary hole entry

Figure – 12 shows injection of air through primary and dilution hole into the liner. It clearly shows the presence of stagnation zone behind swirler. Figure – 13 shows presence of CO formation near swirler exit, Which represents the inadequacy of oxygen availablity from swirler air alone to burn all the fuel completely. With the primary air all these CO is oxidized into CO2, which can be observed from figure - 14.

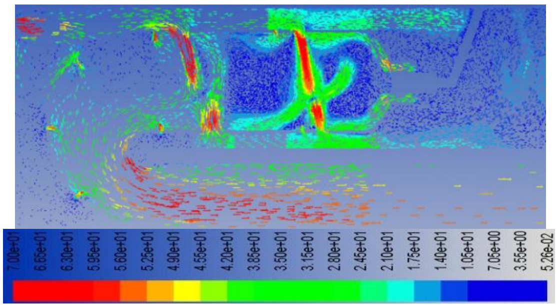

Figure - 12 Mid plane velocity vector plot

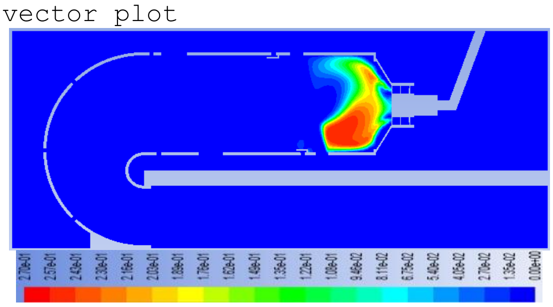

Figure - 13 CO mass fraction contour

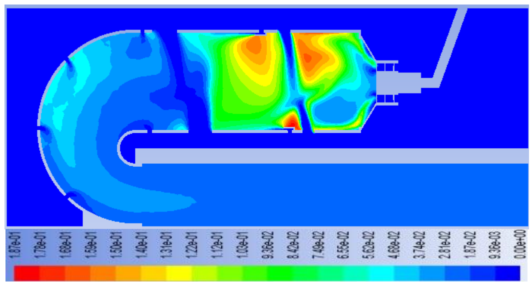

Figure -14 CO2 mass fraction contour

Effect of air to fuel ratio on the performance of combustor

Figure - 15 shows the effect of air to fuel ratio on temperature distribution along the center line of the combustor, with increase in air to fuel ratio high temperature region moves towards the swirler.It clearly shows effect of primary and dilution air jet on temperature distribution. Figure 16,17,18,19 and 20 shows the mid plane temperature contour of 75,100,125,150 and 175 air to fuel ratio cases repectively. Figure 21, 22, 23, 24 and 25 shows the side plane temperature contour of 75,100,125,150 and 175 air to fuel ratio cases repectively. These figures clearly shows the effect of air to fuel ratio on the temperature distribution. With decrease in fuel flow rate high temperature zone moves towards the swirler. At lower fuel flow rates, air coming from swirler is sufficient to burn all the fuel. Figure - 26 shows average teperature at exit for five air fuel ratio cases. Figure - 27 and 28 shows effect of air fuel ratio on circumferential pattern factor and total pressure loss of respectively. Increase in air fuel ratio reduces the pattern factor, because of reduced exit temperature. Increase in air fuel ratio reduces the total pressure loss, because of reduced heat release.

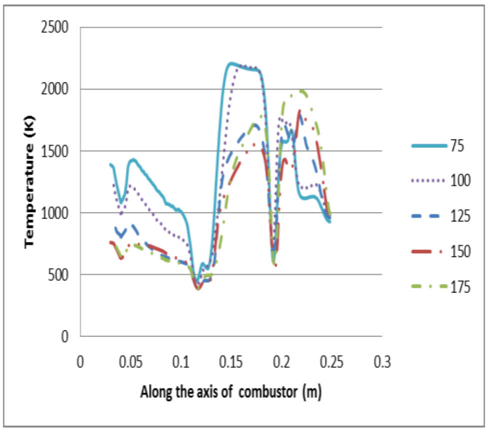

Fig.15 Temperature variation along the center axis of combustor of all five cases of air fuel ratio

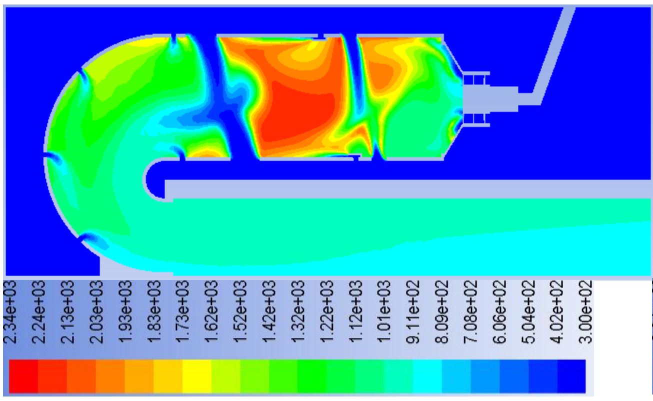

Figure 16 Mid plane temperature contour of AFR 75 case

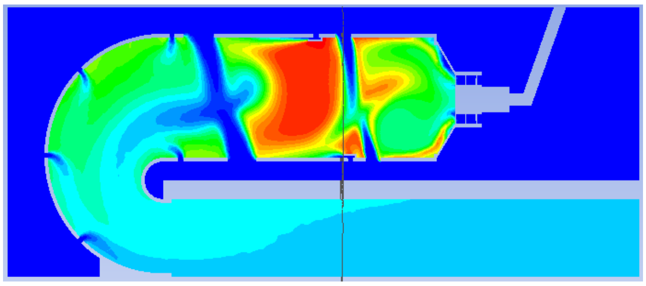
Figure 17 Mid plane temperature contour of AFR 100case

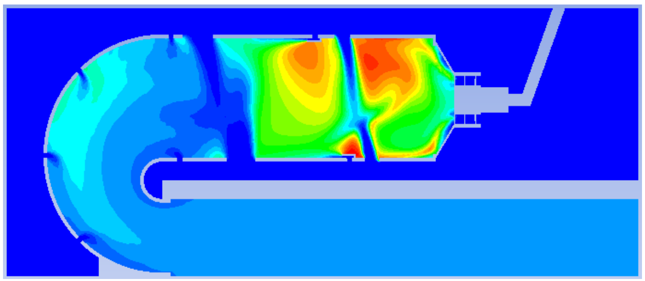
Figure 18 Mid plane temperature contour of AFR 125case

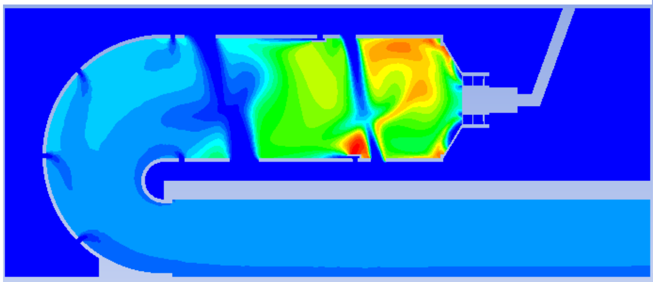
Figure 19 Mid plane temperature contour of AFR 150case

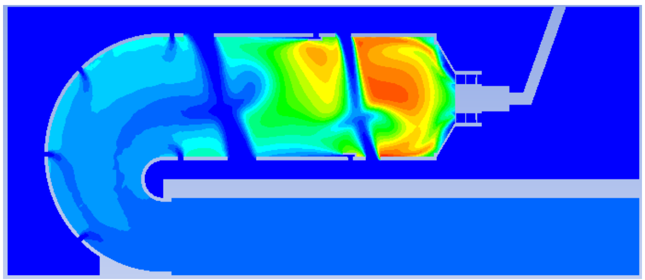
Figure 20 Mid plane temperature contour of AFR 175case

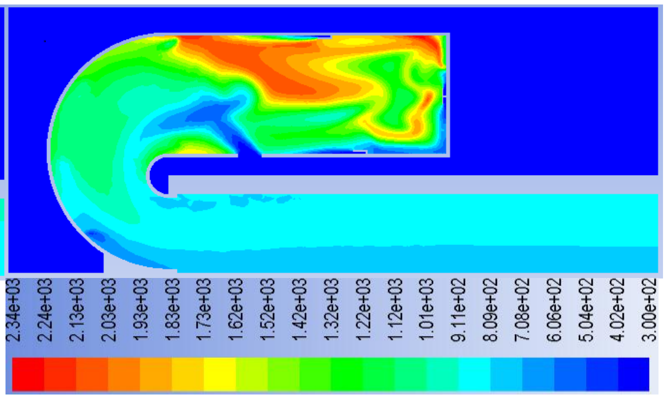

Figure 21 Side plane temperature contour of AFR 75 case

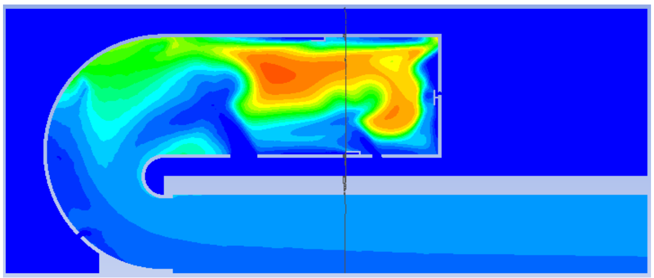
Figure 22 Side plane temperature contour of AFR 100 case

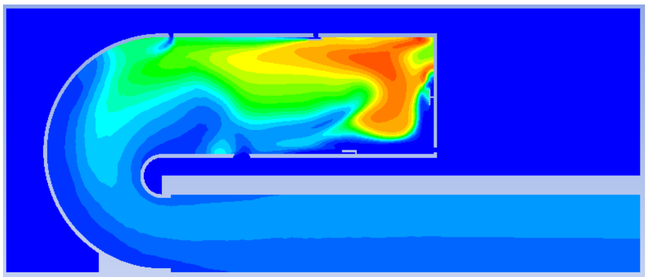
Figure 23 Side plane temperature contour of AFR 125 case

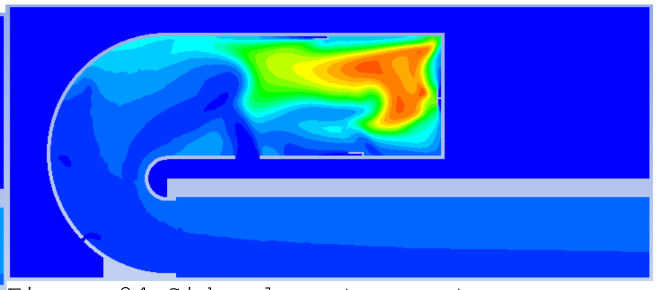
Figure 24 Side plane temperature contour of AFR 150 case

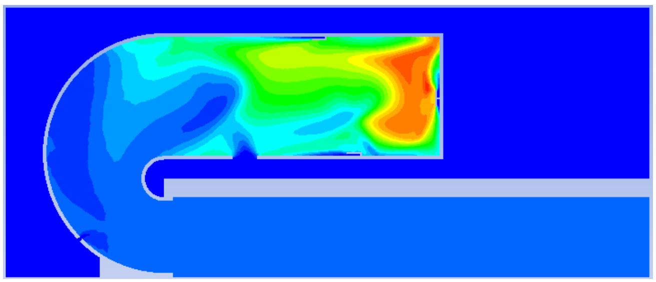
Figure 25 Side plane temperature contour of AFR 175 case

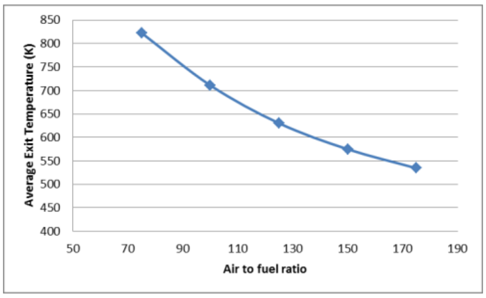

Fig.26 Effect of Air fuel ratio on average exit temperature

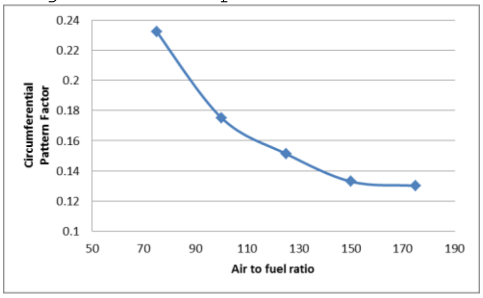

Fig.27 Effect of Air fuel ratio on circumferential pattern factor

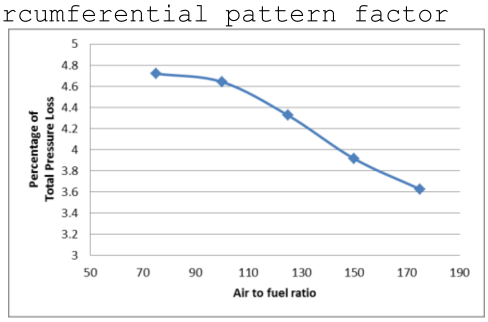

Fig.28 Effect of Air fuel ratio on percentage of total pressure loss

Conclusions

The following conclusions were made from computational study of reverse flow combustor for various values of air to fuel ratios.

1. The cold and hot flow of reverse flow combustor was well captured. It is observed that there is not much change in percentage distribution between hot and cold flow cases.

2. The effect of overall air to fuel ratio on primary zone temperature distribution, total pressure loss and circumferential pattern factor of reverse flow combustor were well understood. It is observed that with increase in air to fuel ratio;

(i) High temperature zone moves towards swirler.

(ii) Exit average temperature decreases

(iii) Pattern Factor decreases, this is because of decrease in exit average tempearture.

(iv) Totatal pressure loss also decresaes due to reduced heat release rate.

Reference

1. D. S. Crocker and C. E. Smith "Numerical Investigation of Enhanced Dilution Zone Mixing in a Reverse Flow Gas Turbine Combustor". J. Eng. Gas Turbines Power 117, 272 (1995)

2. Yufeng Cui, Xuan Lu, Gang Xu, Jianli Chen, ChaoqunNie, and Weiguang Huang"Numerical Investigation of a Stagnation Point Reverse Flow Combustor". ASME Conf. Proc. 2008, 507 (2008)

3.Undapalli; Satish; Srinivasan; Srikant; Menon; Suresh "LES of premixed and non-premixed combustion in a stagnation point reverse flow combustor"Proceedings of the Combustion Institute

4.Dr.Agarwal, Dr.Bharani "Performance evaluation of reverse flow gas turbine combustor using modified analogy"IE(1)Journal-MC